Defense Technical Information Center Compilation Part Notice

ADP023786

TITLE: Modeling Breaking Ship Waves for Design and Analysis of Naval Vessels

DISTRIBUTION: Approved for public release, distribution unlimited

This paper is part of the following report:

TITLE: Proceedings of the HPCMP Users Group Conference 2007. High Performance Computing Modernization Program: A Bridge to Future Defense held 18-21 June 2007 in Pittsburgh, Pennsylvania

To order the complete compilation report, use: ADA488707

The component part is provided here to allow users access to individually authored sections of proceedings, annals, symposia, etc. However, the component should be considered within the context of the overall compilation report and not as a stand-alone technical report.

The following component part numbers comprise the compilation report:
ADP023728 thru ADP023803

Modeling Breaking Ship Waves for Design and Analysis of Naval Vessels

Gabriel Weymouth, Kelli Hendrickson, and Dick K.P. Yue
Department of Mechanical Engineering, Massachusetts Institute of Technology, Cambridge, MA
{khendrk, weymouth, yue}@mit.edu

Thomas O'Shea and Douglas G. Dommermuth
Science Applications International Corporation (SAIC), Naval Hydrodynamics Division, San Diego, CA
{thomas.t.o'shea, douglas.g.dommermuth}@saic.com

Paul Adams and Miguel Valenciano
USACE Engineer Research and Development Center, Major Shared Resource Center (ERDC MSRC), Vicksburg, MS
{Paul.Adams, Miguel.Valenciano}@erdc.usace.army.mil

Abstract

One of the remaining challenges involved in modern naval ship design and analysis is to account for the effects of breaking waves, spray and air entrainment on the performance and non-acoustical signature of a surface ship. The near field flow about a surface ship is characterized by complex physical processes such as: (i) spray sheet and jet formation; (ii) strong free-surface turbulence interactions with (large-amplitude) breaking waves; (iii) air entrainment and bubble generation; and (iv) post-breaking turbulence and dissipation. The challenges associated with this task are twofold. The first is robustly simulating the large-scale problem which involves the flow about an entire surface ship. The second is the development of physics-based closure models for steep breaking waves in the presence of turbulence. To wit, a two-pronged approach consisting of developing an understanding for closure model development and applying cutting-edge computational capabilities has been developed to accurately simulate the free-surface flow around naval combatants.

Using high-resolution direct numerical simulation of the Navier-Stokes equations employeeing the level set method, we have successfully simulated an ensemble of unsteady breaking waves at Reynolds numbers $O(10^{3-4})$. This includes steady and unsteady as well as spilling and plunging events. This dataset is continually being improved upon in terms of depth and breadth as a direct result of this Challenge Project. The goal of this core research area is to develop understanding of the physics of breaking waves to help guide the development of physics-based breaking wave modes. The dataset is being used for the evaluation of closure models for inclusion in current larger scale simulations such as large eddy simulation and Reynolds-Averaged Navier-Stokes.

Robustly simulating the near-field flow of a surface ship requires the development of new models and numerical techniques suitable for use in large scale applications. We have performed more moderate-scale simulations to design, verify, and validate these capabilities before their implementation on the large-scale simulations.

Using Numerical Flow Analysis (NFA), simulations of several naval combatants were performed at a range of speeds. The numerical results show wave overturning at the bow and flow separation at the transom. Air is entrained along the side of the hull and in the rooster-tail region behind the stern. In both regions, numerical predictions agree well with experimental measurements. This work marks the first time that NFA has been used to simulate an entire ship hull. The numerical simulations were performed on the Engineer Research and Development Center (ERDC) Cray XT3 using 128–256 processors. Approximately, 90 million grid points were used in the simulations.

1. Introduction

The Navy's design requirements for faster ships operating in unconventional settings are pushing the envelope of current modeling capabilities. The predictive uncertainties introduced by breaking waves, spray formation and air entrainment have ramifications on the signatures of proposed vessels that in turn limit the

0-7695-3088-5/07 $25.00

confidence in analysis of new designs. The Navy is attempting to incorporate high performance predictive capabilities in their suite of design tools that can reliably predict these key features. This Challenge Project-supported work adopts a two-pronged approach for developing these tools by advancing the state of the art in general by providing robust solvers for complex fluid flows while concurrently developing physics-based models for complex breaking wave and air entrainment phenomena.

A simple engineering analysis demonstrates that these flow phenomena depend on multiple and disparate time and length scales. At the smallest scales, spray formation and the bubbly contact line are dependent on length scales orders of magnitude smaller than the length of the ship with correspondingly small time scales as well. At the largest scales, the overall structures of the shipwake including the unsteady rooster tail and breaking waves along the hull are a function of the beam of the ship, ship speed, etc. Capturing the effects of the very small-scale physics on these large-scale features in a practical design environment is a primary concern in Computational Ship Hydrodynamics. Even with the progress that has been made in computational methodologies and the increase in computational capabilities available to the user, a single simulation which hopes to resolve these phenomena would be a monumental undertaking and could not be completed in the near future.

There is hope that these physics may be simulated directly using massively parallel, large-scale applications with models to bridge the gap between physics and resolvable scales. This requires a robust flow solver capable of accurately simulating complex flows with minimal user guidance which ultimately incorporates physics-based models. The Numerical Flow Analysis (NFA) code provides this turnkey capability to simulate the near-field flow around a surface ship. The Level Set Large Eddy Simulation (LS-LES) provides the framework for model development.

The goal of this work is to use high resolution simulations of the complex physical process involved at their most basic level, to develop necessary models, and to incorporate them into large-scale ship simulations. These results must then be compared with experiments to determine if the models were sufficient. The overall result will be a general, robust numerical capability for use in the design of surface ships in a practical design environment.

2. Problems and Methodologies

Two methodologies specifically designed for the scales involved are used. NFA uses Cartesian-grid and volume-of-fluid methods to simulate the flow around surface ships. The Cartesian-grid portion of the formulation ensures that complications due to gridding are eliminated. The sole geometric input into NFA is a surface panelization of the ship hull which is used to impose no-flux boundary conditions on the hull surface. The volume-of-fluid portion of the numerical algorithm is used to capture the free-surface interface, including large-scale effects of breaking waves, spray formation, and air entrainment. The Euler equations are solved using a domain decomposition method. Processor communication on the Cray XT3 is performed using either Cray's shared memory access library or Message Passing Interface (MPI). The CPU requirements are linearly proportional to the number of grid points and inversely proportional to the number of processors. Together, the ease of input and usage, the ability to model and resolve complex free-surface phenomena, and the speed of the numerical algorithm provide a robust capability for simulating the free-surface disturbances near a ship. Additional details of the NFA formulation are provided in Dommermuth, et al., 2006.

A complementary version of the NFA code was developed at the Massachusetts Institute of Technology (MIT) for use as a test bed for introducing models and improvements to NFA. This version of the code is written in FORTRAN95 and is completely vectorized to maximize performance on high performance computing (HPC) platforms. In addition to these coding improvements, newly developed algorithms are implemented which are formulated in a general way to minimize conditional statements. Examples are the use of a parametric representation for the ship's hull and general analytic body boundary conditions. These improvements can reduce computational costs by an order of magnitude in key subroutines. Further information regarding these algorithms can be found in Weymouth, et al., 2006.

The small-scale application must have many of the features of the large-scale problem. It must robustly handle interface topology changes without *ad-hoc* treatments or human intervention and have very little numerical dissipation which may mask physical dissipation. This feature is highly desirable for developing turbulence models of the micro-scale physics. Until physics-based models are developed, Direct Numerical Simulation (DNS) of the Navier-Stokes equations are required. Once models are developed, LES will be used to further our knowledge of larger-scale problems. To meet this goal, LS-LES was developed by the Massachusetts Institute of Technology. It is a Cartesian grid, Eulerian-interface-capturing scheme based on a modified Level Set (LS) method. Both water and air are treated as incompressible fluids governed by a multi-fluid Navier-Stokes equation. The interface between the two fluids is smoothed over a few grid points for

numerical stability. The Cartesian grid uses a staggered MAC-type formulation; thus no upwinding of the convective terms is necessary. Processor communication is accomplished via MPI making the code portable to any platform. In particular, this code performs well on the IBM P4, the Cray X1, and the Cray XT3. The CPU requirements are linearly proportional to the number of grid points and inversely proportional to the number of processors. Both large-scale and small-scale applications have been validated against experiments and optimized for use on HPC resources through the support of previous Office of Naval Research (ONR) programs and Challenge Projects. Additional details of LS-LES are provided in Hendrickson, 2004.

3. Results & Discussion

3.1. LS-LES

Flow behind a submerged object produces a wave train, one to two body lengths behind the body. Providing the object is near enough to the surface, the first wave of the resulting wave train will form a quasi-steady breaker (Duncan, 1983). DNS of quasi-steady breaking waves is performed by simulating the flow past a submerged object of diameter 1 centered at x=0. An immersed boundary condition method is used to simulate the presence of a submerged cylinder using LS-LES. The key focus of this work is to use the geometry to generate a quasi-steady breaking wave. Thus, viscous effects about the body are not considered and the condition enforced at the body is a no-penetration condition. Figure 1 shows a waterfall plot of the surface elevation for the latest quasi-steady breaking wave simulation for the wavebreaking database. Included also are contours of the transverse vorticity at the air-water interface. The flow is from the left to the right while the body is fixed. Time increases along the vertical profile. This simulation is for a moderate Reynolds number of 800. Surface tension effects are included ($We=\rho u D^2/\sigma=10$) and can be seen in the presence of capillary waves on both wave crests, evident by the bands of vorticity shown in the plot. The motion of the capillary waves on the front of the first wave crest between $1 < t < 2$ represent an initial development of the wave due to the manner in which the body is slowly introduced into the uniform inflow. The primary wave of interest is the second crest in the image, first behind the body. As it forms a bulge and capillary wave train system, it becomes unstable and the wave breaks by the bulge moving down the face of the wave. However, because of the continual source of energy to the flow from the presence of the body, the bulge system begins to set back up again prior to the end of this simulation.

A time history of the vorticity field in both the air and water flows during this event is shown in Figure 2. Prior to the motion of the bulge (Figure 2a), the vorticity field beneath the bulge is essentially a single (negative) vortex beneath the bulge. Near the onset of motion of the bulge, the presence of a small positive vortex can be detected (Figure 2b, x/D~3.2). During the bulge motion, the vorticity field becomes richer in that small vortex-pair structures can be identified as well as the presence of large-scale vortical structures in the wake of the breaking crest. Figure 3 shows a close-up of the bulge region after the event has started. The cyclic motion of the bulge on the front face of the wave has been seen in experiments (Duncan, 1983). For the development of physics-based models, a longer simulation is required to catch the cyclic breaking of the bulge. However, this type of information is critical for the validation and development of physics-based mixed-phase turbulence models for LES and "off-the-shelf" Reynolds-averaged Navier-Stokes capabilities.

3.2. ATHENA in Waves

Predictions of the horizontal and vertical loads on the model hull form of the US Navy Research Vessel ATHENA held fixed in regular head sea waves were made with the NFA. The three-dimensional numerical simulation was performed using 850×192×128= 20,889,600 grid points, 5×8×4=160 sub-domains, and 160 nodes on the Cray XT3 at the ERDC. The length, width, depth, and height of the computational domain are respectively 4.0, 1.0, 1.0, 0.5 ship lengths (L_o). Grid stretching is employed in all directions. The smallest grid spacing is 0.0020 L_o near the ship and mean waterline, and the largest grid spacing is 0.020 L_o in the far field. The Froude number is $Fr = U_o / \sqrt{gL_o} = 0.4136$ where U_o is the ship's speed and g is the acceleration of gravity. Two incident wavelengths (λ) are considered: $\lambda/L_o = 2$ and $\lambda/L_o = 1/2$. In both cases, the wave steepness is $H/\lambda = 0.06$, where H is the wave height. The equations for the wavemaker are imposed ahead of the ship. Initial transients are minimized by slowly ramping up the free-stream current and the incident wave amplitude. For this simulation, the non-dimensional time step is t=0.0005. The numerical simulation runs 10,000 time steps corresponding to 5 ship lengths. Each simulation requires about 80 hours of wall-clock time.

Tests of the model hull form were done at the David Taylor Model Basin (DTMB) in Carderock, MD. Figures 4 and 5 show perspective views of the predicted free-surface elevations around the ship at time t = 5. The predicted and measured free-surface elevations as a function of time at x = 1.5 are shown in Figures 6 and 7. The predicted free-surface elevations are ramped up to their full height. This minimizes transients associated

with starting up the numerical wavemaker. The measured free-surface elevations show slight irregularities that are associated with limitations with the wavemaker at DTMB.

Figures 8 and 9 show the predicted drag force compared to measurements. In the case of the numerical simulations, the drag is initially zero because the model is ramped up to full speed from zero forward speed. The primary harmonics that are evident in the plots are due to the incident wave forces. We speculate that the higher harmonics that are evident in the laboratory results, which are especially evident for the longer wave case, are due to vibrations in the structure that is used to restrain the model. In general, numerical predictions and laboratory measurements agree well.

Figures 10 and 11 show the predicted vertical force compared to measurements. The displacement has been subtracted out from the results. As the model ramps up to full speed, a mean suction force is induced on the model. The oscillatory portion of the force is dominated by hydrostatics. Once again, numerical predictions and laboratory measurements are in good agreement.

It is not possible to show all of the details in the numerical predictions through the use of figures. In order to study the flow in even more detail, several animations have been prepared at the flow visualization center at ERDC. The animations show forced motions of naval combatants, naval combatants in waves, and naval combatants moving with constant forward speed. The animations are viewable at http://www.saic.com/nfa.

Acknowledgements

This research is supported by ONR under contract numbers N00014-04-C-0097. Dr. Patrick Purtell is the program manager. The David Taylor Model Basin under the guidance of Dr. Arthur Reed also supports this research. The numerical simulations have been performed on the Cray XT3 at the ERDC, the Cray X1 at the Army High Performance Computing Research Center, and the IBM P4s at the Naval Oceanographic Office.

References

Brocchini, M. and D.H. Peregrine, "The dynamics of strong turbulence at free surfaces. Part 2: Free-surface boundary conditions." *Journal of Fluid Mechanics*, vol. 449, pp. 225–290, 2001.

Carderock Division, *Surface Ship Model 5415*, 2005, http://www.dt.navy.mil/hyd/sur-shi-mod/.

Dommermuth, D.G., T.T. O'Shea, D.C. Wyatt, M. Sussman, G.D. Weymouth, D.K.P. Yue, P. Adams, and R. Hands, "The numerical simulation of ship waves using cartesian-grid and volume-of-fluid methods." *Proceedings of the 26th Symposium on Naval Hydrodynamics*, Rome, Italy, 2006 (to appear).

Duncan, J.H., "The breaking and non-breaking wave resistance of a two-dimensional hydrofoil." *Journal of Fluid Mechanics*, vol. 126, pp. 507–520, 1983.

Hendrickson, K., "Navier-Stokes Simulation of Steep Breaking Water Waves with a coupled air-water interface." Sc.D. Thesis, Massachusetts Institute of Technology, 2004.

Weymouth, G., D.G. Dommermuth, K. Hendrickson, , and D.K-P. Yue, "Advances in Cartesian Grid Methods for Computational Ship Hydrodynamics." *Proceedings of the 26th Symposium on Naval Hydrodynamics*, Rome, Italy, 2006 (to appear).

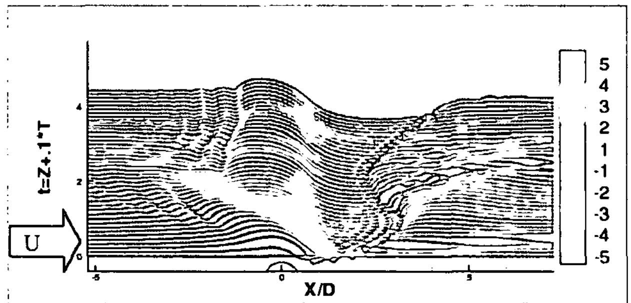

Figure 1. Waterfall plot of surface elevation (colored by vorticity) for quasi-steady breaking wave. The surface profiles are increased by a distance of 0.1 for each T=1.0.

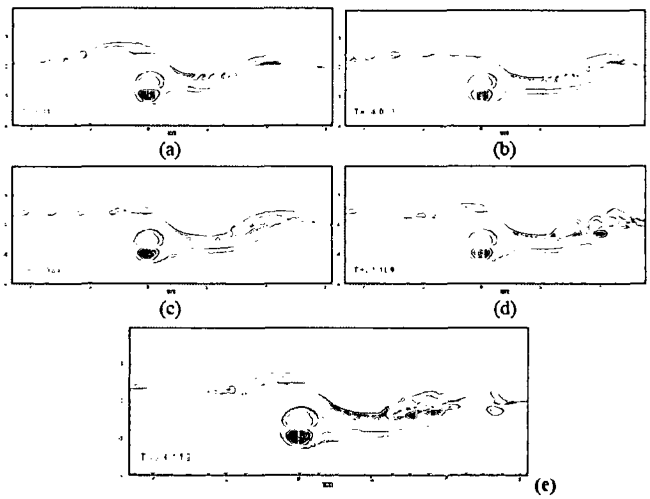

Figure 2. Evolution of vorticity structure in both air and water fluids for a quasi-steady breaking wave. (a) prior to bulge motion; (b) near onset of bulge motion; (c) bulge motion; (d) bulge motion, with large-scale structures; and (e) bulge beginning to set back up.

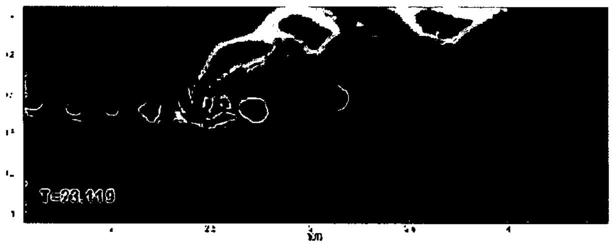

Figure 3. Bulge region after quasi-steady breaking event has begun

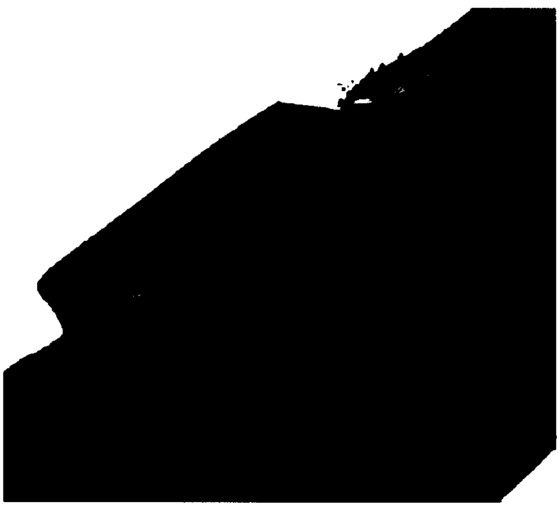

Figure 4. Wave elevation. λ/L_o= 1/2

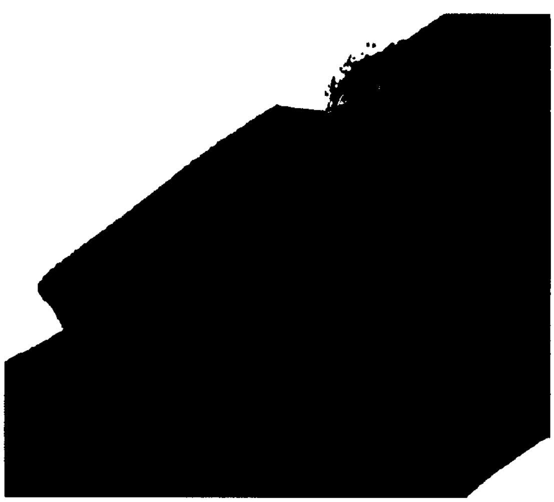

Figure 5. Wave elevation. λ/L_o = 2

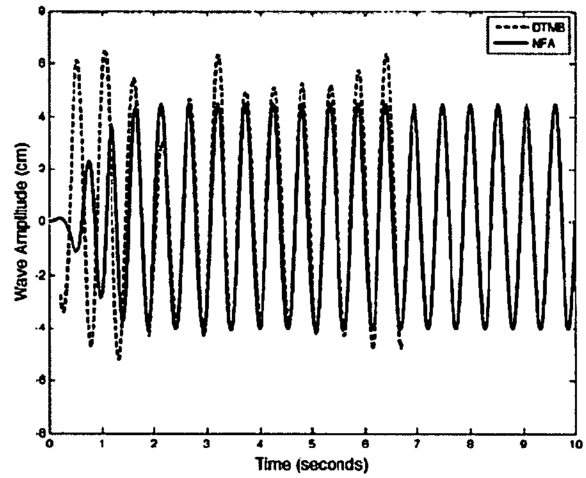

Figure 6. Wave elevation at x = 1.5

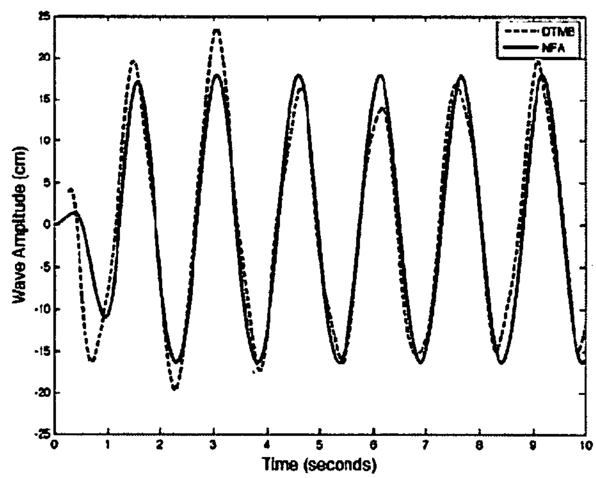

Figure 7. Wave elevation x = 1.5, λ/L_o = 2, λ/L_o = 1/2

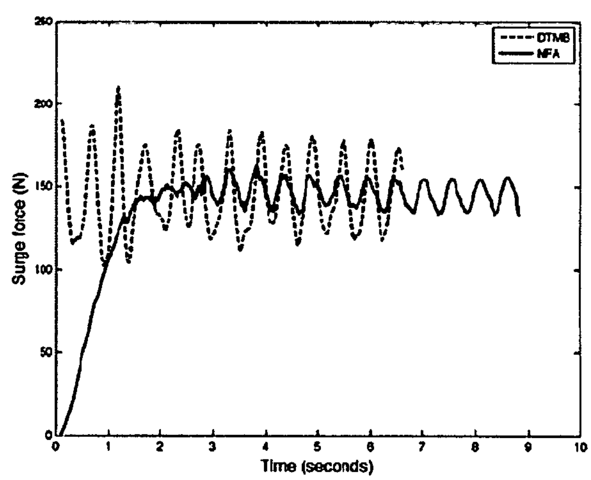

Figure 8. Drag force, λ/L_o = 1/2

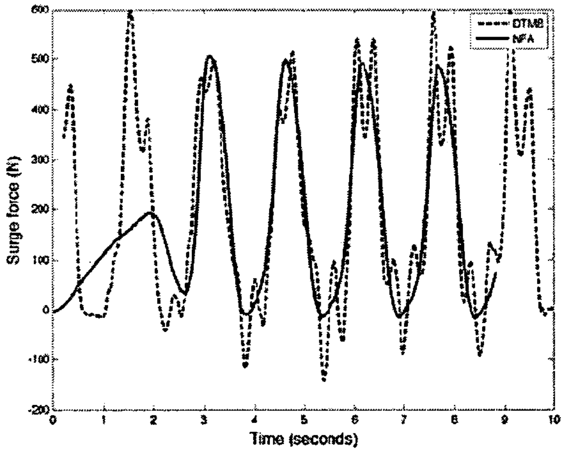

Figure 9. Drag force. $\lambda/L_o = 2$

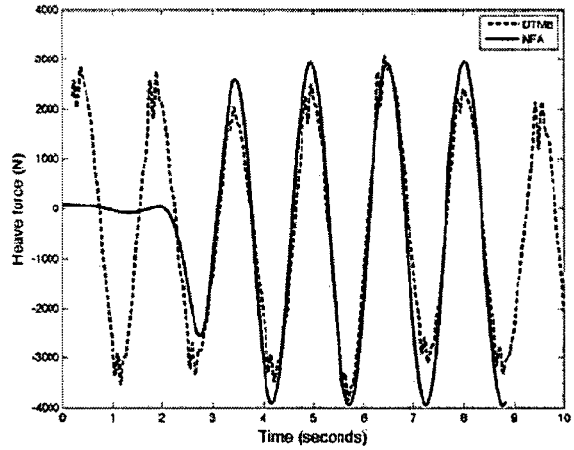

Figure 11. Vertical force, $\lambda/L_o = 2$

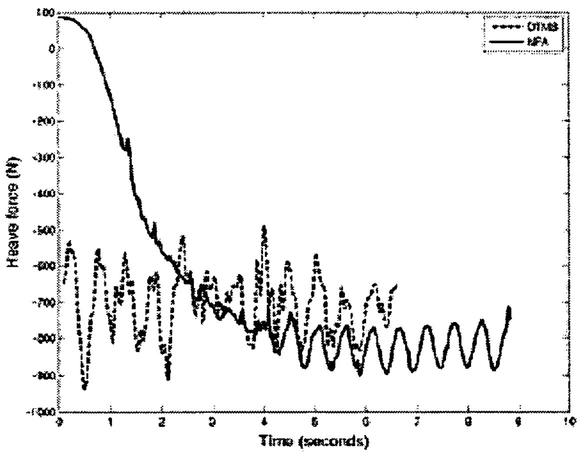

Figure 10. Vertical force, $\lambda/L_o = 1/2$